

EXPLORATION SYSTEMS

The Exploration Systems Mission Directorate is developing new vehicles, capabilities, supporting technologies, and foundational research that will enable sustained human and robotic exploration of the Moon and other destinations beyond low-Earth Orbit. At the heart of NASA's exploration efforts are the Constellation Program's next-generation space vehicles: the Orion Crew Exploration Vehicle that will carry astronauts through space, the Ares I Crew Launch Vehicle that will launch Orion and the crew into orbit, and the heavy-lift Ares V Cargo Launch Vehicle that will carry additional vehicle components and equipment into orbit for rendezvous with Orion. Other key areas of research and technology development include robotic missions to the Moon, health and safety of crews on long-duration space missions, and risk mitigation for exploration projects.

DOUG COOKE

Associate Administrator http://www.nasa.gov/exploration/home/index.html



EXPLORATION SYSTEMS MISSION DIRECTORATE

PRESTON SCHMAUCH
NASA Marshall Space Flight Center
(256) 544-1218
Preston.B.Schmauch@nasa.gov

Close-up of Figure 1

Project Description: Currently, all turbine blade stresses on the Ares I J-2X upper stage engine are determined using blade loading obtained from unsteady computational fluid dynamics (CFD) analyses. The J-2X fuel turbine is a 2½-stage supersonic engine—and there is currently no on-blade data for the second stage of a supersonic engine. To fill this data gap, the Heritage Fuel Air Turbine Test (HFATT) will provide one of the most instrumented air rig turbine tests ever performed. The air rig test runs the turbine at scaled conditions in air as opposed to actual engine conditions. It will utilize substantial on-rotor, unsteady pressure instrumentation and will include various instrumentation on the second stage. This test will help reduce risk for the J-2X turbopumps.

To assess and modify the run box that envelopes all run conditions that the turbine could experience during the upcoming HFATT testing, various test points were simulated. Unsteady CFD simulations were performed to help determine the outer corners of the run box and the best transient locations to investigate. Torque, power, and speeds from the simulations were examined to ensure that the test box was enveloped by the limitations of the test facility at NASA Marshall Space Flight Center (MSFC). In addition, the engine design point pressure ratio, flow coefficient, and various other points were run and interrogated to help produce design curve estimations for different pressure ratios and to formulate detailed pre-test predictions. Temperature and pressure-range data obtained from the unsteady CFD simulations were used to calibrate the various time-accurate instruments that will be used in testing.

One area of particular interest in the HFATT and CFD simulations is unsteady pressure frequencies. Currently, unsteady blade loading is determined by doing a Fourier decomposition of the pressures at every computational node solved in the unsteady CFD simulation. A stress team then uses those coefficient results in a forced-response analysis. Several modes

show low factors of safety for the engine running conditions and are consequently an area of interest in the HFATT. Comparisons will be made to determine any conservatism that might exist in the CFD simulations and will subsequently be used for code validation. NASA is currently having tip dampers employed for some of the J-2X turbine blades to increase factors of safety.

Relevance of Work to NASA: The Ares I J-2X upper stage engine is a critical component in reaching NASA's current goal of returning to the Moon. Due to the higher thrust requirements of the Ares I upper stage, J-2X turbopump running conditions were increased to accommodate the additional thrust. The increased rotational speeds of the J-2X turbopumps have led to increased stresses and lower factors of safety. The HFATT project will increase model fidelity and reduce risk associated with the J-2X engine design and analysis. The information obtained in the testing will also help improve safety and design quality for future programs.

Computational Approach: Phantom, a NASA-developed code that has been anchored and validated for supersonic turbines, is being used for the HFATT unsteady CFD simulations. Phantom uses three-dimensional, unsteady Navier-Stokes equations as the governing equations. The Baldwin-Lomax turbulence model is used for turbulence closure. An overset O and H grid topology with moving grids to model blade motion is employed for the simulations. To simulate several run points in and on the run box, a periodic 1/7 sector of the turbine was modeled for all off-design running points, and a full annulus simulation was run for the design point.

Results: All simulations needed for pre-test support of HFATT have been completed. The run box was sculpted using results obtained from the unsteady CFD analyses, such as those shown in Figures 1 and 2. One example of this is the run conditions at one corner of the run box corresponding

to a low flow rate and low rotational speed. The simulation results showed that this preliminary point would "unchoke," and the run box was altered to avoid this phenomenon. We are currently awaiting the instrumented hardware before testing can begin. Testing is projected to begin in May 2009.

Role of High-End Computing: In total, 23 different run points were simulated and analyzed in support of the HFATT. To run the high number of test points, significant resources were required. Without NASA's HEC resources and the Columbia supercomputer, we would not have been able to provide such a comprehensive level of support within a reasonable time frame. Approximately 2 million processor-hours were needed to complete all simulations for the HFATT project.

Future: Pre-test run conditions do not always match actual running conditions, and the HFATT is no exception. After HFATT testing, test points with actual run conditions will be simulated using NASA HEC resources. These simulations will

provide comparisons to assess the quality of the CFD methods currently used for stress analysis. Alternate test configurations could also be simulated using NASA HEC resources.

Co-Investigators

Daniel Dorney, NASA Marshall Space Flight Center

- [1] Dorney, D., Griffin, L., and Schmauch, P., "Unsteady Flow Simulations for the J-2X Turbopumps," 54th JANNAF Propulsion Meeting, May 2007.
- [2] Marcu, B., Tran, K., Dorney, D., and Schmauch, P., "Turbine Design and Analysis for the J-2X Engine Turbopumps," 44th AIAA Joint Propulsion Conference, *AIAA 2008-4660*, July 2008.
- [3] Marcu, B., Zabo, R., Dorney, D., and Zoladz, T., "The Effect of Acoustic Disturbances on the Operations of the Space Shuttle Main Engine Fuel Flowmeter," 43rd AIAA Joint Propulsion Conference, AIAA 2007-5534, July 2007.
- [4] Dorney, D., Griffin, L., Marcu, B., and Williams, M., "Unsteady Flow Interactions between the LH2 Feedline and SSME LPFP Inducer," 42nd AIAA Joint Propulsion Conference, AIAA 2006-5073, July 2006.

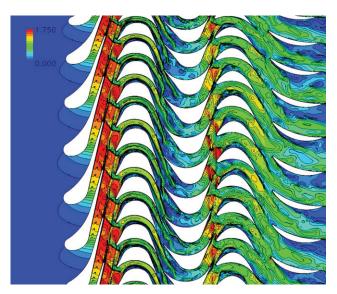


Figure 1: Mach number contours from simulation of the J-2X fuel turbine air rig.

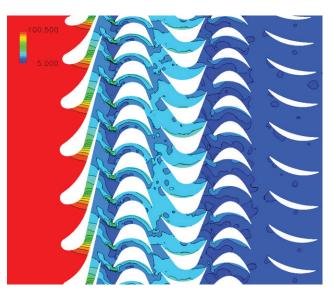


Figure 2: Static pressure contours from simulation of the J-2X fuel turbine air rig (psi).

ARES I ROLL CONTROL SYSTEM JET EFFECTS ON CONTROL ROLLING MOMENT IN FLIGHT

EXPLORATION SYSTEMS MISSION DIRECTORATE

KHALED S. ABDOL-HAMID NASA Langley Research Center (757) 864-8224 K.S.Abdol-Hamid@nasa.gov

✓ Close-up of Figure 1.

Project Description: It is important to determine effects of the roll control system (RoCS) jets on the rolling moment of the Ares I Crew Launch Vehicle. This challenging task includes determining how best to model the chemically reactive jet plumes from the RoCS and how to capture the plume interactions with the freestream airflow around a complex vehicle with many detailed protuberances. Computational fluid dynamics (CFD) simulations help accomplish this task by providing efficient, detailed aerodynamic data to supplement the more limited and costly experimental wind tunnel test data. Several computational analyses have previously revealed the impacts of installing the RoCS on different configurations.

The objective of this project is to quantify jet effects on the net efficiency of the Ares I vehicle's RoCS in flight. These jet effects vary with flow conditions such as flight Mach number, local atmospheric pressure, angle of attack, and the roll angle of the vehicle. Therefore, a practical database construction requires many hundreds of flow computations for a given vehicle outer mold line (OML) and RoCS thruster configuration. The present task is divided into three phases. The first phase included a preliminary study of computational best practices along with examinations of jet flow properties in a cross-flow and roll control thruster plume interaction with the vehicle. An existing computational mesh for an Ares I-X flight test vehicle configuration with a simplified OML was used in the preliminary study. The second phase was a limited parametric study of jet effects for the Ares I configuration at selected Mach numbers, angles of attack, and roll angles. The third phase will be a parametric CFD study of approximately 125 flow conditions along a nominal Ares I ascent trajectory. Results of these computations will provide an aerodynamic database for guidance and control simulation applications.

Relevance of Work to NASA: Complementary to ground-based wind tunnel testing, CFD methods are being used extensively to support design analysis and aerodynamic database

development for NASA's next-generation of space exploration vehicles. The Ares Project has designated the CFD flow solver USM3D as the primary code for developing the computational aerodynamic database for the vehicle. Results of this study have provided critical insight into the process by which jet plumes affect RoCS performance in flight.

Computational Approach: Flow solution development uses the USM3D Navier-Stokes solver on a tetrahedral, unstructured mesh in which the flow variables are all calculated at cell centers. Pre-processing of mesh cell connectivity, partitioning for parallel computing, and flow input conditions are conducted using NASA's HEC resources, as are extensive post-processing steps required to extract information from the solutions for Ares I design applications. The overall computational process is known as the Tetrahedral Unstructured Solver System (TetrUSS). Although the thruster and freestream flow conditions were established at full-scale flight vehicle values, the jet effect CFD computations were performed at wind tunnel Reynolds numbers to conserve computer resources by reducing the mesh size and the number of iterations needed to converge each solution.

Results: The solutions were developed over a highly refined mesh system to capture the interaction of jet plumes with cross-flows, as well as the subsequent flow interactions with the vehicle and its protuberances downstream of the RoCS location. The total mesh contained 70 million cells to resolve relevant flow interaction details. The assessment of jet effects at each flow condition required two solutions: one with thrusters idle and one with thrusters firing for positive or negative roll. The differences between the force and moment coefficients of the two solutions provided a quantitative measure of jet effect on RoCS efficiency in flight. An example of the complex interaction between the freestream flow and the RoCS plumes is shown in Figure 1.

Role of High-End Computing: All of the solutions were computed on the NASA Advanced Supercomputing (NAS) facility's Columbia supercomputer at NASA Ames Research Center. The scale of parallel computing required for this project involved 256 processors for each solution and a significant amount of total computer hours. The USM3D flow solver uses approximately 10 microseconds per cell per iteration on a single processor core, and each solution used 256 processors in parallel to solve the flow equations. Even with an immense demand for resources from this and other high-priority projects within the Agency, NASA's HEC Program provided an exceptional level of support for this project with timely allocation of the supercomputer nodes needed to run several jobs simultaneously, and the runtime hours to enable this project to meet its deadline.

Future: This RoCS jet effects study will be part of the aerodynamics database supporting future Ares I design cycles. To achieve precise and smooth vehicle roll control, a subsequent task will provide jet effect data for a large parametric matrix to support guidance, navigation, and control studies. Continuing HEC resource support will be critical to the successful completion of Ares Project mission objectives. We are planning to complete 400 CFD solutions using the Columbia supercomputer. The resources needed to complete this task are estimated to be 5 million processor-hours.

Co-Investigators

• S. Paul Pao and Karen A. Deere, both of NASA Langley Research Center

- [1] Deere, K.A., Pao, S.P., and Abdol-Hamid, K.S. "A Computational Investigation of the Roll Control System Jet Effects on Rolling Moment of the Ares I-X Clean Configuration," NASA TP 2008 (review and publication pending).
- [2] Deere, K.A., Pao, S.P., and Abdol-Hamid, K.S. "A Computational Investigation of the Roll Control System Jet Effects on Rolling Moment of the Ares I A103 Full Protuberance Configuration," NASA TP 2008 (review and publication pending).

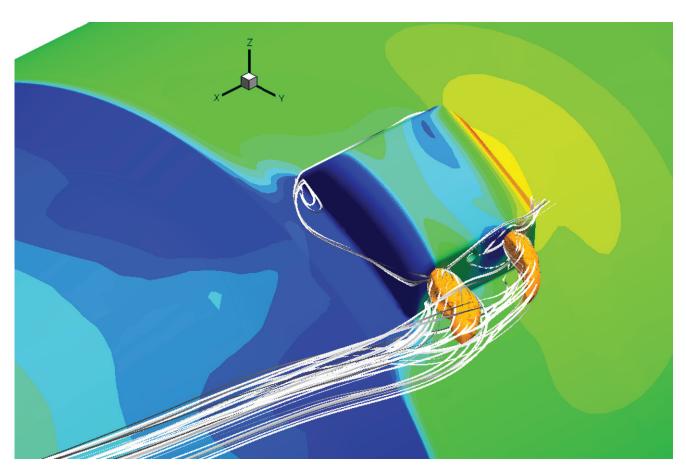
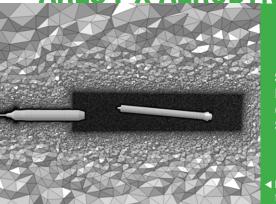


Figure 1: Interaction between the freestream flow and the roll control system plumes on the Ares I Crew Launch Vehicle.

ARES J-X AERODYNAMICS DATABASE DEVELOPMENT



EXPLORATION SYSTEMS MISSION DIRECTORATE

STEVEN BAUER NASA Langley Research Center (757) 864-5946 Steven.X.Bauer@nasa.gov

I Figure 1: Topology of a typical computational grid in the vicinity of the Ares I-X upper stage simulator and the first stage.

Project Description: As a crucial step in the development of the new Ares I Crew Launch Vehicle (CLV) design, NASA will be launching the Ares I-X Flight Test Vehicle (AIX FTV) in 2009. The Ares I-X FTV will be similar in mass and size to the actual Ares I CLV, and will imitate the first two minutes of the Ares I launch trajectory and flight conditions. The test vehicle will launch through a speed of Mach 4.7 and separate from its first stage (FS) solid rocket booster (SRB) at 130,000 feet. The core objectives of this first test flight are to demonstrate performance of the flight control systems, and to characterize the flight environment during stage separation. In support of these efforts, extensive computational fluid dynamics (CFD) simulations have been performed to generate and/or populate databases of aerodynamic conditions for the Ares I-X test flight configurations and parameters. These aerodynamic databases are being used by the Ares I-X Guidance, Navigation & Control (GN&C) community to help assess aerodynamic performance and flight control functionality of the AIX FTV.

Relevance of Work to NASA: This project is an important part of the Agency's development of the Ares I CLV. The extensive simulation data will help supplement and interpret the experimental data obtained from the actual AIX test flight to better evaluate and understand aerodynamic factors of the Ares I CLV design.

Computational Approach: A majority of the CFD simulations for this project were conducted using USM3D (Figure 2), a Reynolds-averaged Navier-Stokes code for unstructured grids developed at NASA Langley Research Center. A second CFD code, OVERFLOW, was also used to perform simulations of first stage descent cases. Grid sizes ranged from 40 million grid cells for the simplest "clean" configurations at wind tunnel Reynolds number conditions, to 90 million grid cells for configurations including the full set of protuberances at flight Reynolds numbers.

Results: Hundreds of CFD solutions (such as that shown in Figure 1) have been generated to build the aerodynamic

databases needed to assess key aspects of Ares I-X flight performance. For the main part of the launch trajectory before first-stage separation, simulation cases covered 11 Mach numbers from 0.9–4.5, angles of attack from 0°–90°, and roll angles from 0°–360°. Various combinations of these parameters were simulated for a simplified, clean version of the vehicle geometry and for a more detailed version of the geometry including protuberances. A number of specific protuberance cases were run on a simplified configuration to determine pressure coefficients and heating rates for various instruments. Selected cases were also run at both wind tunnel condition Reynolds numbers, and full flight condition Reynolds numbers.

CFD simulations were also performed to analyze separation of the AIX vehicle's first-stage (FS) rocket booster from the upper stage simulator (USS), and descent of the FS and USS components as they fall back to Earth. Forty-two stage separation cases were conducted at Mach 4.5 at various separation distances and orientation angles of the FS and USS components. Both USS and FS descent cases were run at Mach numbers from 0.5–4.5 with angles of attack from 0°–180°.

Additional sets of cases were also generated to assess plume effects and control authority for the vehicle's roll control system (RoCS), booster deceleration motors (BDMs), and booster tumble motors (BTMs). Thirty-three RoCS simulations were conducted for three Mach numbers at various roll angles. These cases were run with RoCS not firing, with clockwise RoCS jet pairs firing, and with counterclockwise RoCS jet pairs firing to determine the interference effects of RoCS plumes on the overall generated forces and moments for the vehicle. Twenty-four BDM and two BTM firing cases were conducted for Mach 4.5 at various pitch and roll angles to determine plume effects on the vehicle and the effects of one or more motor "out" cases.

Results of the CFD simulations for all these cases have provided extensive data on the forces, moments, surface pressure coefficients, line loads, and in some cases, heating rates for various conditions of the AIX FTV.

Role of High-End Computing: HEC resources were used to predict forces and moments, surface pressures, line loads, and some heating rates on the AIX FTV. These resources were also used to visualize the flowfield around the vehicle at various flight conditions and around various control system jets as they are firing.

While the majority of computations were run on the Columbia supercomputer, recent work has been moved to the Pleiades system. Extensive efforts to improve the performance of USM3D on Columbia were undertaken by USM3D code developers and NASA Advanced Supercomputing facility staff, and a similar effort is currently underway to improve USM3D efficiency on Pleiades. Typical runs on either system used from 128 processors for the smallest grids to 256 processors for the largest grids. Computations required as few as 12,000 iterations to converge for steady-state solutions on clean configurations, and as many as 180,000 iterations for

time-accurate solutions on configurations with extensive flow separation, translating to runtimes from 11 hours to as many as 160 hours.

Future: Aerodynamic database generation will continue in order to complete all of the RoCS, BDM, BTM, and main engine firing cases for the AIX FTV. Additional cases will also be run to fill in gaps where no data exists or where required by GN&C to evaluate and refine control of the vehicle. After launch of the FTV in 2009, additional cases will be needed to match exact flight conditions of interest, which will be used to assess the veracity of the computational results.

Co-Investigators

 Steven Krist, William Compton, Craig Hunter, and Karen Deere, all of NASA Langley Research Center

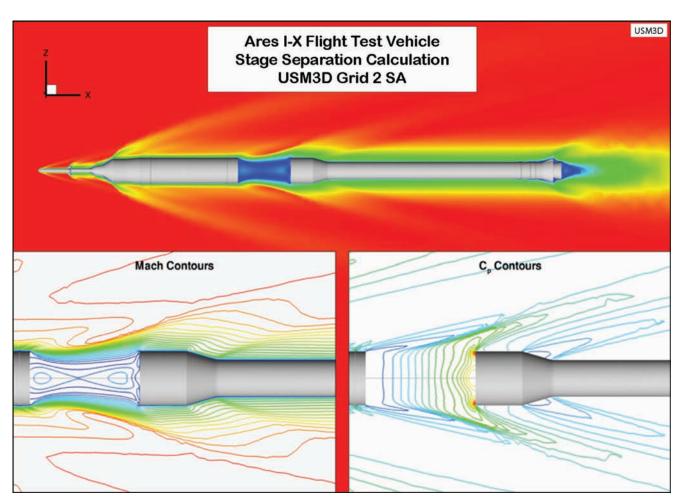
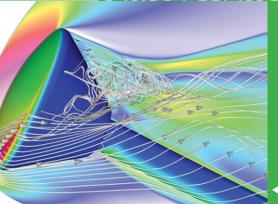


Figure 2: Typical USM3D flowfield for a stage separation case.

CREW EXPLORATION
VEHICLE AEROSCIENCES PROGRAM



EXPLORATION SYSTEMS MISSION DIRECTORATE

JOSEPH OLEJNICZAK NASA Ames Research Center (650) 604-3252 Joseph.Olejniczak@nasa.gov

Figure 1: Simulation of the Orion crew module and wind tunnel sting.

Project Description: The Crew Exploration Vehicle (CEV) Aerosciences Program (CAP) is developing complete aerodynamic and aerothermodynamic databases for the Orion crew module (CM) and launch abort system (LAS) covering the range of all possible operating conditions.

Accurate aerodynamic data such as lift, drag, pitching moment, and dynamic stability derivatives are required to design the flight control system and ensure that the pinpoint landing requirement can be met. The aerodynamic database covers the entire CEV operational envelope including nominal ascent, ascent abort scenarios, on-orbit plume environments, reentry flight from the hypersonic through subsonic regimes, and the terminal landing approach including parachute deployment.

The aerothermodynamic database covers the portion of atmospheric flight that produces significant aeroheating for the vehicle. The ascent heating environment must be quantified to ensure vehicle integrity during nominal and off-nominal ascent conditions. Thermal protection system (TPS) design requires convective and radiative heating environments for the entire vehicle surface during reentry, including localized heating rates on penetrations and protuberances.

The CAP databases are built using a combination of computational fluid dynamics (CFD) results and wind tunnel data, which together provide higher-fidelity databases at lower cost than either could alone. The databases will require thousands of high-fidelity numerical solutions modeling the flowfield around the CM and LAS for all flight regimes. The CFD solutions are also critical to understanding how to extrapolate wind tunnel data to extreme flight conditions that cannot be replicated by ground tests. Conversely, ground test data are also used to help quantify the uncertainty of the CFD solutions. CFD is also used to assess local geometric features, such as the compression pads that attach the crew module to the launch vehicle. For previous programs, heating rates on such features would be determined from an extensive set of ground

test data that would be extrapolated to flight conditions using engineering models. For Orion, however, a small set of ground test data was obtained to quantify the uncertainty in the CFD; and CFD solutions were used to develop the engineering heating model that the designers could use.

Relevance of Work to NASA: The CAP aerodynamic and aerothermodynamic databases are critical to the design and operation of Orion—a key component of NASA's Vision for Space Exploration and Constellation Program objectives. The databases will be provided to the Orion prime contractor as Government Furnished Material (GFM) and will be used to both design and operate the vehicle. These databases are the largest GFM components of the Orion Project and represent a significant investment of NASA resources.

Computational Approach: We use a number of high-fidelity codes to compute the flowfield around the CM and LAS. Using multiple, independent codes for the same flight conditions increases our confidence in the computed results. The DPLR and LAURA codes are reacting Navier-Stokes solvers that include thermochemical nonequilibrium. These codes are used to compute aerothermodynamic heating rates and aerodynamic coefficients in the hypersonic regime. The NEQAIR radiation solver is a first-principles physics code that computes production of radiation by gas in the hot shock layer, transport of photons through the shock layer, and radiative heating to the CEV surface. Aerodynamic coefficients in the subsonic, transonic, and supersonic regimes are computed using four different CFD tools. The OVERFLOW and USM3D codes solve the Reynolds-averaged Navier-Stokes equations using multiple overset structured grids. Cart3D is an inviscid, compressible flow analysis package that uses Cartesian grids to solve flow problems over complex geometries such as the Orion LAS with abort motor (AM) and attitude control motor (ACM) plumes. The unstructured Euler CFD code FE-LISA is also being used.

Results: CAP has generated thousands of three-dimensional CFD solutions for the CM and LAS geometries. These solutions, ranging from Mach 0.3 ascent abort conditions to Mach 40 reentry conditions, have been used to populate multiple aerodynamic and aerothermodynamic databases supporting Orion design analysis cycles and the Orion Preliminary Design Review (PDR). Two specific examples of our simulations are given below.

Figure 1 shows a DPLR CFD calculation of the flow around the CM with the wind tunnel sting. The surface is colored with pressure contours, and streamlines are shown to help visualize the flow. In this example, aeroheating data on the front of the CM helps validate the CFD results for flight conditions, and the CFD quantifies the effect of the sting on the wake flow and afterbody heating—allowing more accurate extrapolation of the wind tunnel data to flight conditions.

Figure 2 shows snapshots of a time-accurate OVERFLOW simulation of the LAS with four of the ACMs firing and the plumes flowing around the LAV. The surfaces are colored by pressure, and the image at lower-left shows the pressure change due to the ACM plume interference effects. Predicting the aerodynamics of the LAS requires accurate modeling of the ACM plumes over a range of freestream Mach numbers from 0–6, a range of angles-of-attack from 0–180 degrees, and various combinations of ACM thrust levels from each of the eight nozzles.

Role of High-End Computing: While each individual solution may only take a few hundred to a few thousand processor-hours depending on the analysis code and geometric complexity modeled, the thousands of high-fidelity CFD solutions needed to populate the CEV databases could not be completed without access to a supercomputer such as

Columbia. The availability of such high-end computing resources, coupled with advancements in CFD fidelity, have allowed the CEV Project to create databases using computational results and provide data complimentary to wind tunnel testing. In addition, high-fidelity simulations of geometrically complex configurations provide insights into flow physics and vehicle response that cannot be obtained with any other method.

Future: Through Orion development and flight operations, CAP will compute thousands of high-fidelity numerical solutions to populate the aerodynamic and aerothermodynamic databases over the next few years. As detailed design work is completed, the geometric models of the Orion vehicles will become increasingly more complex and the computations will become more memory-intensive and time-consuming. Terabytes of complete flowfield data will have to be analyzed and stored for future use.

Co-Investigators

- Stuart Rogers, NASA Ames Research Center
- Benjamin Kirk, NASA Johnson Space Center
- · Richard Thompson, NASA Langley Research Center

Collaborating Organizations

 NASA Ames Research Center, NASA Johnson Space Center, and NASA Langley Research Center

- [1] Hollis, B., et al., "Aeroheating Testing and Prediction for Project Orion CEV at Turbulent Conditions," AIAA Paper No. 2008-1226, January 2008.
- [2] Amar, A., et al., "Protuberance Boundary Layer Transition for Project Orion Crew Entry Vehicle," AIAA Paper No. 2008-1227, January 2008.
- [3] Bibb, K., et al., "Aerodynamic Analysis of Simulated Heat Shield Recession for the Orion Command Module," AIAA Paper No. 2008-0356, January 2008.

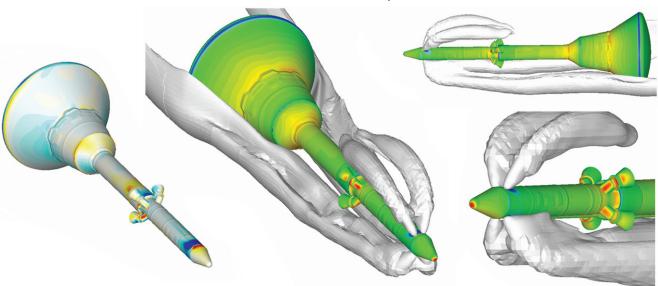
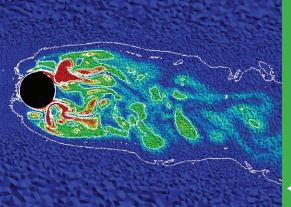


Figure 2: Time-accurate simulation of attitude control motor jets for the Orion Launch Abort Vehicle.





EXPLORATION SYSTEMS MISSION DIRECTORATE

ROBERT BARTELS NASA Langley Research Center (757) 864-2813 Robert.E.Bartels@nasa.gov

■ Detail of Figure 1.

Project Description: For any vehicle on the launch pad, a poorly quantified dynamic ground wind loads environment can result in excessive and potentially catastrophic motion of the launch vehicle. For this reason, the NASA Langley Research Center Aeroelasticity Branch has been tasked with providing the Ares I Project with computational data quantifying the dynamic ground wind environment around a flexible vehicle on the pad. The Ares computational aeroelastic (CAE) analysis team is performing simulations of the Ares I Crew Launch Vehicle (CLV) both on the ground and during ascent. Analyses include static aeroelastic increments for Guidance, Navigation & Control (GN&C), flutter, buffeting, and ground wind loads. In particular, the CAE team performed simulations of ground wind-induced oscillation for both a flexible Ground Wind Loads (GWL) checkout model and the Ares I-X flight test vehicle on a launch pad. The goal of this computational analysis was to provide data to confirm that the proper wind tunnel model environment was achieved.

For a flexible launch vehicle the size of the Ares CLV, accurate wind tunnel-to-flight scaling in the presence of ground winds is difficult because the majority of the dynamic forcing is due to ground wind-induced vortex shedding—an unsteady flow phenomenon where vortices are created at the back of a body and detach periodically from either side. Vortex shedding about a smooth cylindrical body is sensitive to Reynolds number, especially within the critical Reynolds number range. Outside of the critical range the shedding pattern has a welldefined Strouhal frequency, but within this range the shedding is chaotic. The wind tunnel and the flight scale Reynolds numbers for this vehicle fall within the low and high ends of the critical Reynolds number range, making wind tunnel data difficult to scale to full-size flight conditions. Additionally, there are inherent uncertainties in the structural vibration properties of any wind tunnel model, especially for a ground turntable mount system like that used in the wind tunnel test for this case. To help account for these uncertainties, we performed computational aeroelastic simulations of the ground wind environment surrounding the flexible launch vehicle. The checkout wind tunnel model was used to develop best practices and calibration data for subsequent wind tunnel testing of Ares configurations. Nine wind tunnel conditions at which peak bending moments were observed for the checkout model, and the baseline test condition for the Ares I-X vehicle were simulated. The computed unsteady pressures, tie-down bending moments, and shedding frequencies were compared with wind tunnel data.

Relevance of Work to NASA: This work supports the Ares Program and is important to achieving a successful launch of NASA's next-generation space exploration vehicles. The computational data from this project have provided a benchmark against which wind tunnel data can be compared, and have reduced the uncertainty in wind tunnel and empirical design data, resulting in a better vehicle design.

Computational Approach: The Navier-Stokes computational fluid dynamics (CFD) code FUN3D was used to simulate the static and dynamic flexible vehicle responses to ground wind-induced oscillation. FUN3D is an unstructured, finitevolume node-based CFD code developed by the Computational Aerosciences Branch at NASA Langley. The simulations were performed with a one-equation turbulence model and with a Detached Eddy Simulation (DES) turbulence model. FUN3D is capable of performing distributed Message Passing Interface (MPI) parallel computation and was compiled using the libraries Metis, ParMetis, myapich, and the Intel compiler. The launch pad was modeled as a symmetry plane with computational grids containing 10-40 million grid points. Turbulent and laminar flowfields with dynamic fluid/structure interaction (FSI) were simulated using an algorithm that includes dynamic, time-accurate deformation of the volume grid. The structural model used was an MSC. Nastran finiteelement model.

Results: The GWL checkout model simulation confirmed the levels of vehicle motion observed in the wind tunnel tests. It also provided insight into the shedding mechanism that was

producing high tie-down bending moments. The checkout model simulations showed that strong, coherent shedding patterns occurred mainly over the crew module and first stage rocket. However, the simulations also showed that the largest bending moment dynamics occurred in conditions for which there were the largest levels of shedding dynamics over the crew module and launch abort tower regions. Because of this result, the Ares I-X wind tunnel model was populated with additional unsteady pressure transducers in the crew model region. Figures 1 and 2 show sample vorticity contours from the simulations, and Figure 3 shows an example of resulting surface pressures on the vehicle.

Role of High-End Computing: Computations for this work were performed on the NASA Advanced Supercomputing (NAS) Division's Columbia supercomputer, with each simulation distributed over hundreds of processors. Because the vortex shedding of a full-scale vehicle occurs in a critical Reynolds number range, these computations were exceedingly challenging, requiring hundreds of processors and typical runtimes of one to four weeks. Data post-processing was performed using Tecplot on the NAS platform.

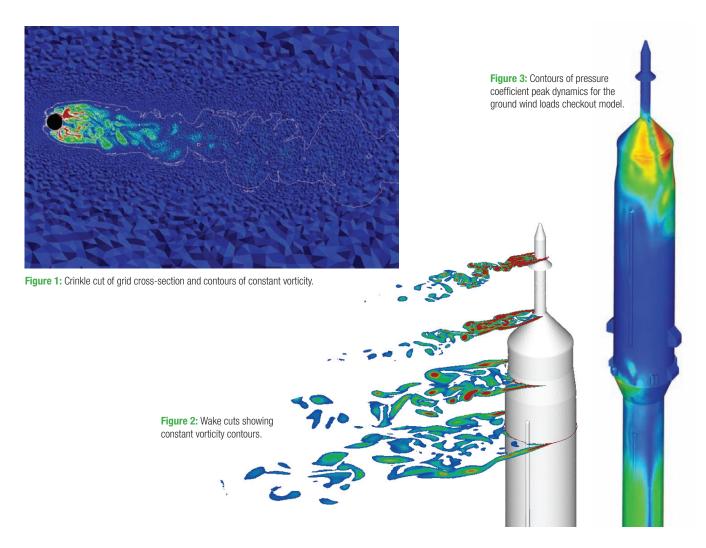
Future: Ares computational aeroelastic analyses will continue to quantify the flexible response of the Ares I and Ares V vehicles. Ascent static analysis will provide flex-to-rigid increments for Guidance, Navigation and Control development. Flutter and buffet analyses will be performed to ensure that the vehicle retains structural integrity throughout ascent. Ground wind loads analyses will continue to further reduce wind tunnel data uncertainties.

Co-Investigators

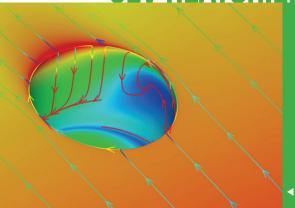
 Pawel Chwalowski, Ray Mineck, Robert Biedron, and Steve Massey, all of NASA Langley Research Center

Publications

[1] Bartels, R. E., "Development of Advanced Computational Aeroelasticity Tools at NASA Langley Research Center," NATO RTO Specialists Meeting on Advanced Aeroelasticity AVT-154, paper 003, May 3-6, 2008.



COMPUTATIONAL SUPPORT FOR ORION CEV HEATSHIELD TPS DESIGN AND ANALYSIS



EXPLORATION SYSTEMS MISSION DIRECTORATE

MICHAEL J. WRIGHT NASA Ames Research Center (650) 604-4210 Michael.J.Wright@nasa.gov

Close-up of Figure 2

Project Description: A thermal protection system (TPS) will protect NASA's Orion Crew Exploration Vehicle (CEV) as it returns from space. In order to understand the extreme environments that the TPS must withstand, aerothermal computational fluid dynamics (CFD) is used to simulate both ground tests and the actual flight environment. NASA's CEV Aerosciences Program (CAP) is responsible for generating aerothermal databases to be used for sizing the TPS system for Orion, while the TPS Advanced Development Project (TPS-ADP) is responsible for simulating high-energy TPS tests and damaged TPS materials. The primary goals of the TPS-ADP are to develop two heat shield concepts to be presented at the Orion TPS downselect review in spring 2009, and to recommend a single design to the CEV Project Office. Aerothermal analyses of high-energy arc jet tests and the effects of in-orbit micrometeorite damage are key components of the overall risk and reliability assessment of the TPS system.

Arc jets are powerful, high-energy facilities used for testing TPS materials in environments similar to those encountered during atmospheric entry. The largest NASA arc jet can operate at a rated power of 60 megawatts per kilogram of gas for up to an hour at a time to simulate extreme entry conditions. The TPS-ADP conducts testing in arc jet facilities at NASA Ames Research Center and NASA Johnson Space Center, as well as in a Department of Defense facility in Tullahoma, Tennessee. Because of the high energy levels and great degree of thermo-chemical non-equilibrium in these tests, analysis is required to understand these facilities and the experimental results. Therefore, CFD simulations are combined with measured data to determine important flow quantities that cannot be ascertained from experiment alone. These simulations are a vital step in both designing the experimental program and understanding how materials perform in intense, chemically reacting flows. Results from the testing and analysis help to drive material selection and overall heatshield design, and will form a cornerstone of the TPS selection.

The potential effects of TPS damage due to micrometeoroid orbital debris (MMOD) strikes are simulated using CFD and material response codes. The required analysis includes: high-velocity impact testing of the TPS materials to determine what types of damage can occur after an MMOD impact; aero-thermal CFD analysis to determine the heating augmentation that would result from this type of damage; and ablative material response analysis to determine whether the damaged TPS would still maintain the required internal temperatures.

Relevance of Work to NASA: This work enables both experimental support and assessment of localized damage effects that are pertinent to the design and materials selection for the CEV heat shield. The CEV is NASA's next-generation vehicle for human space operations, both for missions to low-Earth orbit and to eventually carry astronauts to the Moon and return them safely to Earth.

Computational Approach: Aerothermal environments are simulated using the Data Parallel Line Relaxation (DPLR) hypersonic real-gas flow solver developed at NASA Ames. Researchers from Ames' Space Technology Division have developed best practices for use of both the parallel DPLR code and supporting utility codes to accurately characterize chemically reacting arc jets and test specimens. Such simulations rely heavily on experimental measurements for boundary conditions. These simulations are geometrically simple but physically complex, requiring the simultaneous solution of up to 17 coupled partial differential equations, finite-rate chemistry, and thermo-chemical non-equilibrium. A representative arc jet simulation, such as the one shown in Figure 1, may use upwards of 180 processors for up to 5 hours, and several of these solutions are often necessary to match the experimental measurements.

The DPLR code is also used for orbital debris damage simulations, which are split between larger capsule-only runs, and local damage site runs. This process leverages existing techniques that have been used regularly on the Columbia supercomputer during Space Shuttle missions. Each typical orbital debris damage simulation may require over 15 to 20 hours of computing using up to 200 processors. Figure 2 shows a simulation of the full CEV with small-scale damage to the shoulder region.

Results: Analysis results from this project have contributed to the last three design cycles of the CEV spacecraft, as well as to the documentation delivered to Lockheed Martin as part of the transition of primary responsibility for heat shield development. The TPS-ADP is currently on track to deliver two heat shield design concepts for Phenolic Impregnated Carbon Ablator and Avcoat TPS materials, including an MMOD damage risk assessment for each, for the Orion Preliminary Design Review (PDR).

Role of High-End Computing: The NASA Advanced Supercomputing (NAS) facility supercomputers are used for the CFD simulations of damaged TPS materials—the most numerically intensive part of these analyses. The availability of the Columbia supercomputer was an important contributor to the TPS design team's success.

Future: The TPS Advanced Development Project will continue to use HEC resources to support the final TPS downselection in the spring of 2009. The TPS Insight/Oversight team will use HEC resources to independently verify Lockheed Martin analysis results between the PDR and the Critical Design Review.

Note: On April 7, 2009, NASA selected the Avcoat ablator system for the Orion crew module.

Co-Investigators

 Todd White, Mike Barnhardt, Tahir Gökçen, Dinesh Prabhu, all of ELORET Corporation

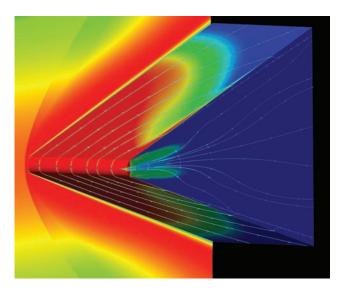


Figure 1: Data Parallel Line Relaxation (DPLR) simulation of a thermal protection system sample wedge in an arc jet flow.

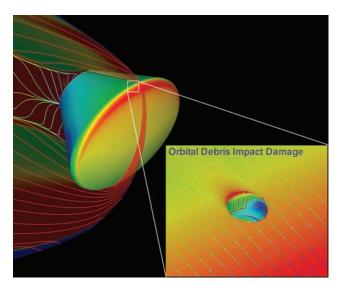


Figure 2: DPLR simulation of the full Crew Exploration Vehicle, with small-scale damage to the shoulder region.

J-2X FUEL AND OXIDIZER TURBINE SIMULATIONS INCLUDING DISC AND TIP CAVITIES

PREST NASA I (256) 5 Prestor

EXPLORATION SYSTEMS MISSION DIRECTORATE

PRESTON SCHMAUCH

NASA Marshall Space Flight Center (256) 544-1218 Preston.B.Schmauch@nasa.gov

✓ Close-up of Figure 1.

Project Description: The Ares I upper stage turbopumps operate in a highly dynamic environment that can be very sensitive to small changes in the flow path. The heritage J-2 turbines were designed with tip seals that minimized flow between the main flow path and the disc and tip cavities. The current J-2X configuration, however, has the tip seals removed and the main flow path is free to interact with any flows in the cavities. Our objective is to simulate the current J-2X turbine configurations and determine what performance and fluid interaction effects will result from the removal of the tip seals.

To better understand the flow effects that will result from removing the tip seals in the current configurations, we are performing full-annulus, unsteady computational fluid dynamics (CFD) simulations of the oxidizer and fuel turbines, both with and without cavities. The simulations must be run full-annulus because flow in the cavities is not a periodic phenomenon. By comparing fluid property histories between the simulations with and without cavities, we will better understand the magnitude of the impact the design change will have on the performance and life of the parts. The results will also be used to increase our current understanding of the conservatisms that are built into modeling and analysis methods.

Relevance of Work to NASA: This project directly affects the Ares I vehicle by reducing risk associated with the J-2X engine and, thereby, the upper stage. Its results will help NASA and contractors make more educated decisions on possible design modifications for current turbopump configurations. These simulations could also reveal benefits or areas of concern that might not be considered otherwise. Knowledge from these simulations will also be applied to other engines and models used in NASA programs.

Computational Approach: Phantom, a NASA code that has been anchored and validated for supersonic turbines, is

being used for the unsteady CFD simulations. Phantom uses three-dimensional, unsteady Navier-Stokes equations as the governing equations. The Baldwin-Lomax turbulence model is used for turbulence closure. An overset O and H grid topology with moving grids to model blade motion is employed for the simulations. With all blades modeled in these simulations, more than 65 million computational nodes were used on the turbine alone. During this project, the Phantom code was also adapted to allow for the addition of the multiple, attached tip and disc cavity grids necessary for these simulations.

Results: Simulations of the J-2X oxidizer turbine with and without cavities were recently completed. Preliminary post-processing has shown significant blade-loading and dynamic pressure environment differences between the cavity and non-cavity cases. Figure 1 shows surface pressure contours from simulation of the J-2X oxidizer turbine with cavities. Preliminary stress analysis shows an increase in the factor of safety on stress when running with the cavities attached. It is believed that the cavities reduce the highly unsteady blade-loading due to the row-to-row blade passing, and convert that energy into a more broadband unsteadiness. More analysis is underway to investigate other implications this effect could have on stress margins. The J-2X fuel turbine model currently has the first disc cavity added and the simulation is still being worked.

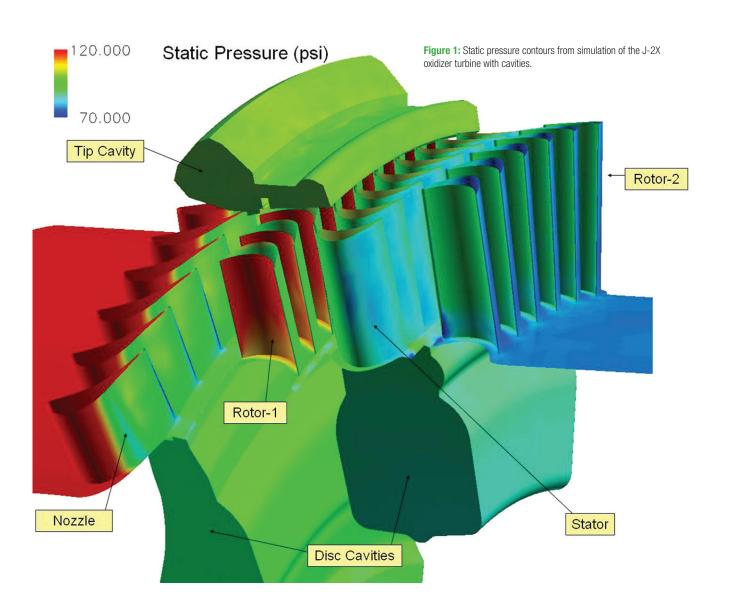
Role of High-End Computing: The size of the simulations for this project required a minimum of about 400 gigabytes of memory. The J-2X oxidizer turbine simulation with cavities alone required 256 processors on the Columbia supercomputer running for approximately 200 days, or 1.2 million processor-hours. More than 2 million additional processor-hours were used to simulate various power balances and configurations of the turbines without the cavities. Without the highend computing resources at the NASA Advanced Supercomputing (NAS) facility, we would not have had the capability to run these simulations.

Future: The computational grids for the J-2X fuel turbine with cavity are currently being generated and will be simulated using NAS HEC resources. It is expected that the results of these simulations will call for further analyses with refined focus on specific topics. Currently, old power balances are being used for the cavity simulations so that a one-to-one comparison can be done for the case without cavities. Future analyses may be done with updated power balances.

Co-Investigators

• Daniel Dorney, NASA Marshall Space Flight Center

- [1] Dorney, D., Griffin, L., and Schmauch, P., "Unsteady Flow Simulations for the J-2X Turbopumps," 54th JANNAF Propulsion Meeting, May 2007.
- [2] Marcu, B., Tran, K., Dorney, D., and Schmauch, P., "Turbine Design and Analysis for the J-2X Engine Turbopumps," 44th AIAA Joint Propulsion Conference, AIAA 2008-4660, July 2008.
- [3] Marcu, B., Zabo, R., Dorney, D., and Zoladz, T., "The Effect of Acoustic Disturbances on the Operations of the Space Shuttle Main Engine Fuel Flowmeter," 43rd AIAA Joint Propulsion Conference, AIAA 2007-5534, July 2007.
- [4] Dorney, D., Griffin, L., Marcu, B., and Williams, M., "Unsteady Flow Interactions between the LH2 Feedline and SSME LPFP Inducer," 42nd AIAA Joint Propulsion Conference, AIAA 2006-5073, July 2006.



PROXIMITY AERODYNAMICS OF THE ARES I LAUNCH VEHICLE DURING STAGE SEPARATION MANEUVERS



Project Description: The objective of this project is to quantify proximity effects on the aerodynamics of the Ares I Crew Launch Vehicle (CLV) during inflight separation maneuvers such as rocket booster stage separation and crew capsule separation in the course of an emergency abort. This can be challenging because the proximity effects vary with flow conditions such as flight Mach number, local atmospheric pressure, angle of attack, and Reynolds numbers. In addition, proximity effects are strongly dependent on the relative positions of the separating components of the CLV, such as separation distance, relative angle of attack, and lateral offsets. Computational fluid dynamics (CFD) analyses are used along with wind tunnel data to develop aerodynamic databases for use in designing the Ares I launch vehicle. A practical database construction requires many hundreds of computations for a given launch vehicle configuration and set of flow conditions.

This project was divided into several phases. First, a preliminary study of crew capsule separation during aborts using Apollo-based configurations established computational best practices and examined the flow properties of the proximity effects. In the second phase, extensive numerical simulations of several types of stage separation scenarios were generated to provide data for a staging trade study conducted at NASA Marshall Space Flight Center. The third and ongoing phase is a parametric study of proximity effects on the Ares I configuration, with the staging process selected by the Ares Staging Trade Study group. The next phase will involve a similar parametric study with the inclusion of plume effects for the upper stage settling motors, booster separation motors, and upper stage main engine.

Relevance of Work to NASA: Results of this study have provided critical insight into the proximity effects on the aerodynamic performance and risk assessment of the Ares I launch vehicle. The proximity aerodynamic data with plume effects that will be generated for a large parametric matrix will

support Guidance, Navigation, and Control (GN&C) and risk assessment studies to achieve a precise and safe stage separation process.

Computational Approach: Most of the flow simulations were conducted on the Columbia supercomputer using the OVERFLOW Navier-Stokes solver with overset structured meshes. Both steady-state and time-accurate simulations were required, depending on the separation distances and other parameters. For small separation distances between the stages, the flowfield was typically steady. At larger distances, however, flows were usually unsteady and time-accurate simulations were required. In addition, inviscid simulations were also conducted using the Cart3D Cartesian mesh generation code to provide guidance in establishing grid resolution requirements for the viscous OVERFLOW simulations. Prescribed motion and fully coupled six-degrees-of-freedom (6-DOF) simulations were also used when appropriate.

Results: The numerical simulations provided important data for the Staging Trade Study group and enabled the Constellation Engineering Management Council to select and finalize the stage separation process to be used for the CLV. The numerically derived data provided both guidance for setting up wind tunnel tests and data impossible to obtain from wind tunnel tests, such as proximity effects at very small or very large separation distances. Figure 1 shows a typical result from the staging trade study effort, capturing the interaction of the J-2X upper stage main engine plume with the interstage (IS) as it falls through the main plume. The concern for this type of stage separation was whether the IS would collide with the J-2X nozzle bell.

Role of High-End Computing: The scale of parallel computing required for this work—involving upwards of 256 processors for each solution, grid sizes of 100–200 million grid points, and runtimes of up to 20,000 processor-hours per case for the

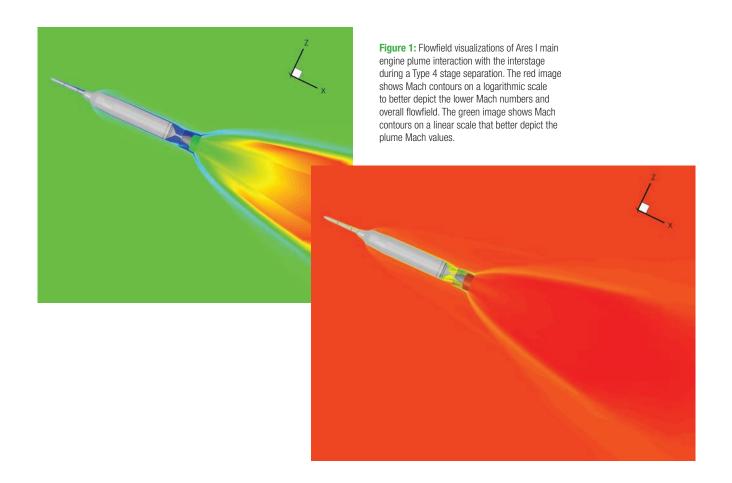
steady-state problems—was quite large by production CFD standards. The NASA Advanced Supercomputing (NAS) Division provided an exceptional level of support for this project, with timely allocations of Columbia supercomputer nodes needed for running several jobs simultaneously, and computing hours needed to meet project deadlines.

Future: The next phase of this project will be to study plume effects on proximity aerodynamics. This data will be part of aerodynamic database generation in support of future Ares I design cycles. The computing requirements will ramp up substantially due to increased grid resolution requirements and the increased number of parameters involved. Continuing use of NAS high-end computing resources will be critical for achieving the Ares I Project mission objectives.

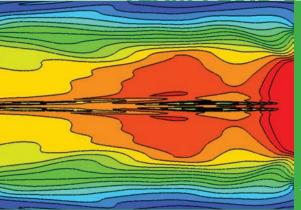
Co-Investigators

- Jeffrey Onufer, Shishir Pandya, and William Chan, all of NASA Ames Research Center
- Veronica Hawke-Wong, ELORET Corp.

- [1] Chan, W.M., Klopfer, G.H., Onufer, J.T., and Pandya, S.A. "Proximity Aerodynamics Analyses for Launch Abort Systems," AIAA Paper 2008-7326, 26th AIAA Applied Aerodynamics Conference, Honolulu, Hawaii, August 2008.
 - * Most of this work is International Traffic in Arms Regulations (ITAR)controlled and is not published.



THRUST OSCILLATION FOCUS TEAM FLUID DYNAMICS ANALYSIS SUPPORT



EXPLORATION SYSTEMS MISSION DIRECTORATE

PHILIP DAVIS
NASA Marshall Space Flight Center
(256) 544-7728
Philip.A.Davis@nasa.gov

◄ High-resolution close-up of Figure 2.

Project Description: A major issue facing the Ares I Crew Launch Vehicle (CLV) is the presence of thrust oscillations that are predicted to occur at the vehicle's acoustic resonance frequency, potentially causing problems for the vehicle and the astronauts aboard. NASA engineers are currently investigating strategies to reduce or eliminate the thrust oscillations. This project aims to assist those engineers by examining the internal flow dynamics of the Space Shuttle Reusable Solid Rocket Motor (RSRM) to learn the cause of the thrust oscillations in the current four-segment RSRM and determine how these problems can be corrected for the future five-segment booster.

Computational fluid dynamics (CFD) techniques are being applied to RSRM geometries at burn times of 80 and 110 seconds using Phantom, a NASA-developed, three-dimensional flow solver. These times were chosen because they exhibit the greatest thrust oscillations in data acquired from the firing of test motors and past shuttle motors. This study centers on the vortex shedding that occurs downstream of the three inhibitors that protrude into the flow within the motor. The number of vortices at each location, the frequency of creation, and the relative strength of each vortex is under investigation, as well as how the vortices affect the acoustics of the motor. Figures 1 and 2 show a shadowgraph and flow velocity contours for the 80-second RSRM burning profile. Additionally, fluid dynamics analysis is being performed at NASA Marshall Space Flight Center's Cold Flow Testing Facility. This facility has been used in the past to test the internal acoustics of the RSRM using a 10% scale model with air as the fluid. The current CFD analysis focuses on past testing performed at the facility, and will be used to help determine what testing should be done in the future. Figure 3 shows a scaled entropy plot of the Cold Flow Testing Facility main chamber.

Relevance of Work to NASA: This analysis will provide NASA with insight into their current technology, including the

performance of the Space Shuttle RSRM and how to perform future testing inside the NASA Marshall Cold Flow Testing Facility. It is expected that this information will help to resolve the thrust oscillation problems currently facing the Ares I vehicle.

Computational Approach: Phantom is a fully three-dimensional, unsteady, finite-difference code that is third-order accurate in space and second-order accurate in time. It utilizes the general equations set for liquids and gases, and is capable of handling two-phase flows. The turbulence model is a modified Baldwin-Lomax model. The code also has the ability to use A*pⁿ boundary conditions for burning surfaces.

Results: The first simulations for this project were begun in December 2007. Inhibitor heights were adjusted to correct levels in March 2008, and corrected head-end burning was implemented in June 2008. In July 2008, the project began collecting data for analysis with the final iteration of geometries. Currently, simulations have generated one second of data for each of the cases that are still running. Four simulations for the Cold Flow Facility geometry are running with varying flow conditions to attempt to match heritage data.

Role of High-End Computing: Each of the RSRM geometries consists of 31,130,190 computational grid cells, which are divided into 120 passages run in parallel on 120 processors using the Message Passing Interface (MPI) library on the Columbia supercomputer. This system has been vital and necessary to achieving the grid density needed to provide the best analysis possible, as even using 120 processors per simulation only accumulates approximately 0.025 seconds of data in 24 hours. At least 2 seconds of CFD data must be generated for the RSRM simulations, which would be impossible without the computing resources that have been provided by the NASA Advanced Supercomputing Division.

Future: The current CFD simulations for the RSRM geometries are not yet complete and will continue to accumulate data. The Cold Flow Facility CFD simulations are still at the beginning stages of data collection and will continue to run to accumulate more data. Once these CFD simulations are completed, the next phase of analysis could include assessing mitigation strategies for reducing thrust oscillations. One example of this would be to alter the inhibitor heights at one or several of the joint locations. Other analyses that could be performed include using the Ares I solid booster geometry as well as future Cold Flow Facility setups.



